HEIDENHAIN

TNC 320

The Compact Contouring Control for Milling, Drilling and Boring Machines

October 2009

HEIDENHAIN
Manual operation
TOOL TABLE
ENT
END

Contents

The TNC 320...

Where can it be used?	Compact and Versatile – The right control for milling, drilling and boring machines	4
How does it look?	Well Designed and User Friendly – The TNC 320 in dialog with the user	6
What can it do?	Minimize Setup and Nonmachining Time – The TNC 320 makes setup easy – The TNC 320 manages and measures	8
	Machining with Four Axes – Swivel head and rotary table controlled by the TNC	12
How is it programmed?	Programming, Editing, Testing – The TNC 320 opens endless possibilities – Graphic support in any situation	14
	Programming in the Workshop – Straightforward function keys for complex contours – Programming contours unconventionally – Field-proven cycles for recurring operations – Reusing programmed contour elements – Fast availability of all information	16
	Open for Communication – Fast data transfer with the TNC 320 – The TNC 320 programming station	22
Are there any accessories?	Workpiece Setup and Measurement – Setup, presetting and measuring with touch trigger probes	24
	Tool Measurement – Measuring length, radius and wear right on the machine	25
	Positioning with the Electronic Handwheel – Delicate axis traverse	26
... At a glance	Overview – User functions – Specifications – Options – Accessories	27

Compact and Versatile
– The Right Control for Milling, Drilling and Boring Machines

For more than 30 years, TNC controls from HEIDENHAIN have been proving themselves in daily use on milling, drilling and boring machines, and machining centers. While the controls have undergone continuous development during this period, the basic operational technique has remained the same.

You will find these principles implemented in the TNC 320 as well: shop-oriented programmability with graphic support, many field-proven cycles and an operational design familiar from other HEIDENHAIN controls.

Shop-oriented programming
You program conventional milling and drilling operations yourself at the machine, in plain language dialog—the workshop-oriented programming language from HEIDENHAIN. The TNC 320 provides you with optimum support with practical prompts, questions and graphical aids.

Standard operations and even complex applications are on call as a large variety of real-world machining cycles or coordinate transformations.

Simple operation
For simple work, such as face milling, you need not write a program. With the TNC 320 it is just as easy to operate the machine manually by pressing the axis keys or—for maximum sensitivity—with the electronic handwheel.

Offline program creation
The TNC 320 can be programmed remotely just as well. Your Fast Ethernet interface guarantees very short transfer times, even of long programs.

The TNC 320 is compact and easy to read.
The TNC 320 is a compact but versatile contouring control for three servo axes and servo spindle. A further servo axis is an option. Thanks to its flexible operating concept—workshop-oriented programmability with HEIDENHAIN conversational programming or offline programming—and its scope of features, it is especially suited for use on universal milling, drilling and boring machines for the following:

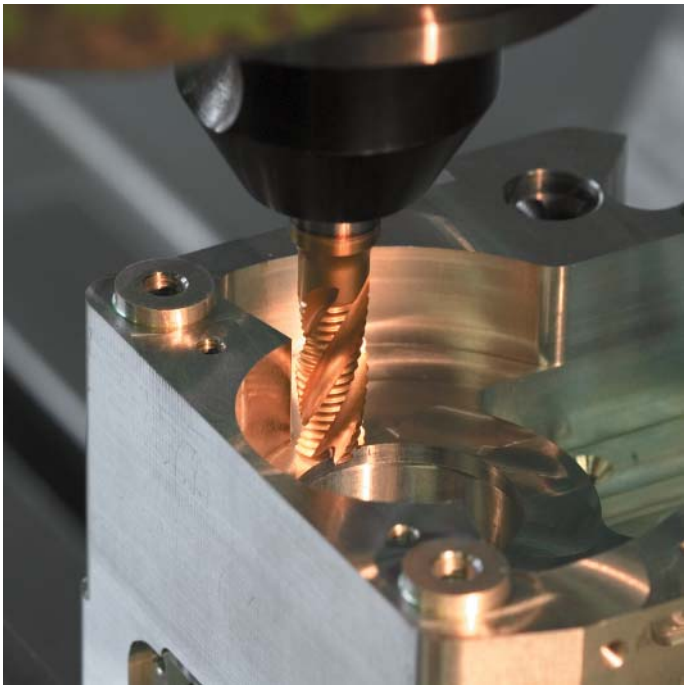
- Series and single-part production
- Toolmaking
- Machine building
- Research and development
- Prototypes and pilot plants
- Repair departments
- Training and education facilities

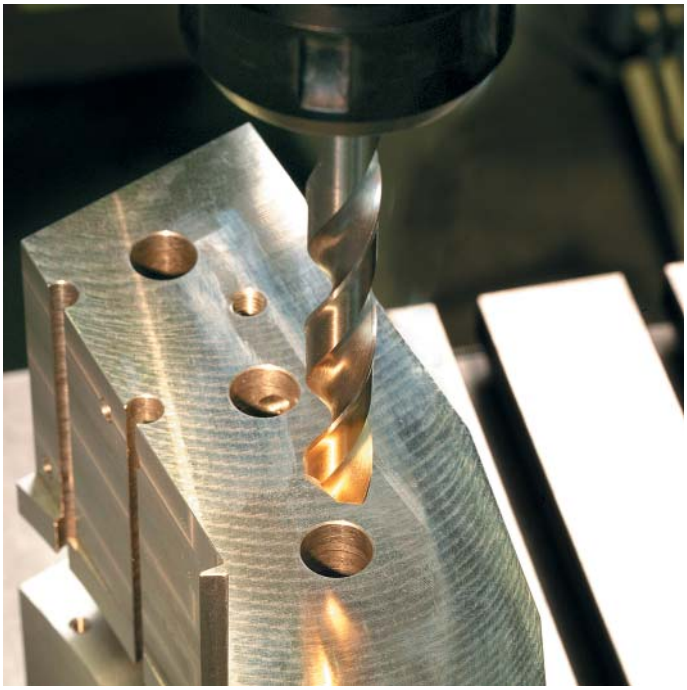
And it offers the applicable features both necessary and helpful for:

Universal milling machines
- Free contour programming
- Milling cycles for complex contours
- Fast presetting with HEIDENHAIN touch probes

Drilling and boring machines
- Cycles for drilling, boring and spindle alignment
- Cycles for linear and circular point patterns
- Drilling oblique holes

Machines with parallel secondary axes
- Compensating movement in the secondary axis U, V, W through the principal axis X, Y, Z
- Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
- Including movements of parallel axes in the position display of the associated principal axis (sum display)



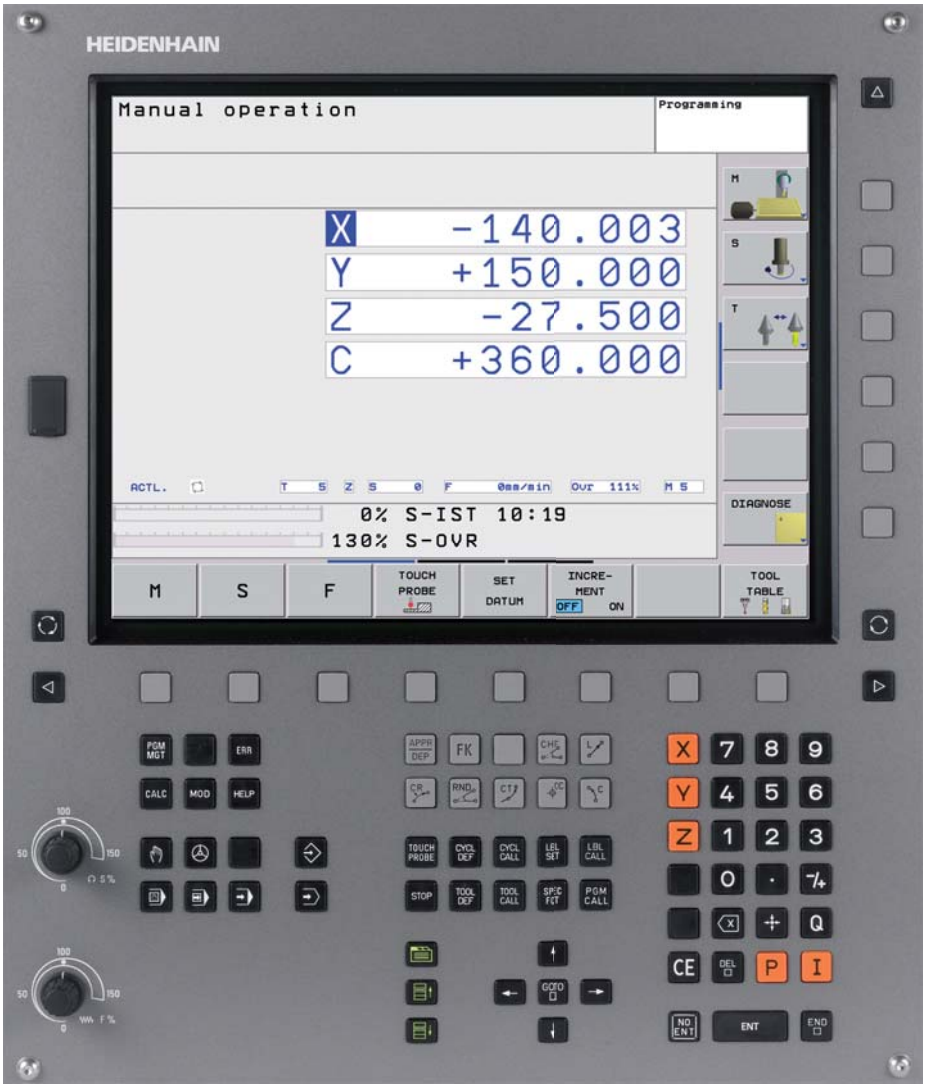


Well Designed and User Friendly
– The TNC 320 in Dialog with the User

The monitor
The TFT 15-inch color monitor shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control, such as program blocks, comments and error messages. More information is provided through graphic support during program entry, test run and actual machining.

The selectable "split screen" display shows the part program blocks in one half of the screen and the graphics or the status display in the other half.

During the course of the program, status displays will always offer information on tool position, the current program, active cycles and coordinate transformations, and other data. The TNC 320 even shows the current machining time.



The keyboard
As with all TNCs from HEIDENHAIN, the keyboard is tailored to the programming process. The well thought-out configuration of keys facilitates program input. Simple words and abbreviations or unambiguous symbols clearly indicate each key's function. Certain functions of the TNC 320 are available by soft key.

Keys on the monitor

Select the screen layout

Display machine mode or programming mode

Soft keys for selecting functions on screen

Shift between soft-key rows

Keys on the control panel

Program/file management, TNC functions

PGM MGT — Program management: Manipulate and delete programs

MOD — Supplementary operating modes

HELP — Help function

ERR — Display error messages

CALC — Show pocket calculator

Machine operating modes

Manual Operation

Electronic Handwheel

Positioning with Manual Data Input

Program Run, Single Block

Program Run, Full Sequence

Navigation and input

Navigation in dialogs

Delete the last entered character

Programming modes

Programming and Editing

Test Run with graphic simulation

L / CHF — Straight line, chamfer

CC / C — Circular arc with center point

CR — Circular path with known radius

CT — Circular path starting tangentially

RND — Corner rounding

APPR DEP — Contour approach and departure

FK — Free contour programming

P — Enter polar coordinates

I — Enter incremental dimensions

Q — Entry of parameter instead of fixed value/Definition of parameter

Transfer actual position to program

TOOL DEF / TOOL CALL — Definition and calling of tools

CYCL DEF / CYCL CALL — Definition and calling of cycles

LBL SET / LBL CALL — Labeling and calling of subroutines and program repeats

PGM CALL — Programmable program call

STOP — Programmed stop, interrupt/ discontinue

TOUCH PROBE — Touch probe functions

SPEC FCT — Special functions, e.g. comments, structure

Unambiguous keys make programming easy.

Many functions are entered by soft key.

With the gray path function keys and conversational guidance, you program line segments and circular arcs defined in various ways.

Spindle speed and feed rate are easily adjusted.

Minimize Setup and Nonmachining Time
– The TNC 320 Makes Setup Easy

Before you can begin machining, you must first clamp the tool and set up the machine, find the position and orientation of the workpiece on the machine, and set the workpiece reference point. Without support from the control this is often a time-consuming procedure, but it is indispensable. After all, any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as for very large workpieces, setup times become quite a significant factor.

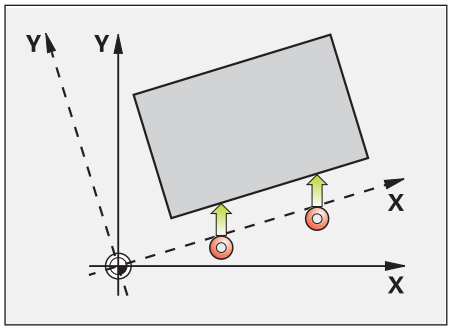
Here the TNC 320 shows its strengths: With its practice-oriented setup features it supports the operator and helps to reduce nonmachining time. Together with the **touch probes,** the TNC 320 offers various probing features for aligning, presetting, and measuring the workpieces.

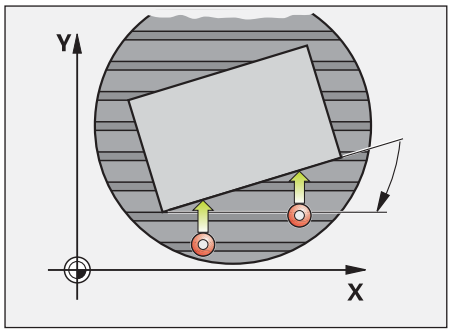
Delicate manual traverse
For setup, you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 26). Particularly with the portable handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

Workpiece alignment
With HEIDENHAIN touch probes (see page 24) and the probing functions of the TNC 320, you can forgo any tedious manual alignment of the workpiece:

- Clamp the workpiece in any position.
- The touch probe ascertains the workpiece misalignment by probing a surface, two holes, or two studs.
- The TNC 320 compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment.

Compensating workpiece misalignment
Compensate misalignment by rotating the coordinate system or turning the table





Workpiece presetting
You can use a reference point to assign a defined value in the TNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy.

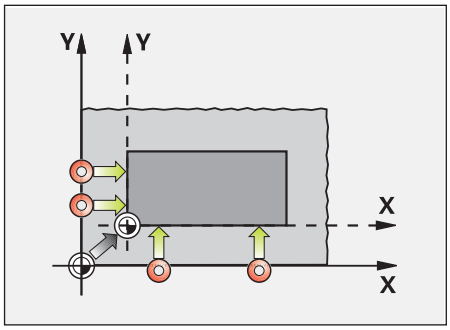
The TNC 320 features probing cycles for automatic presetting. Once found, you can save reference points
- in the workpiece preset table,
- in a workpiece datum table, or
- by directly setting the displayed value.

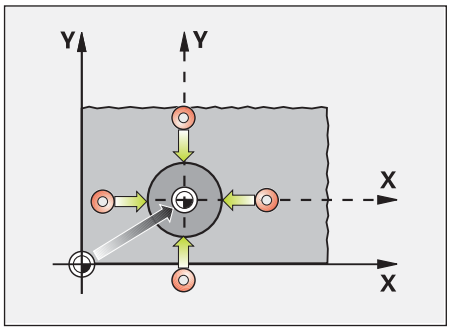
Preset table: The TNC 320's central reference point management
The preset table makes flexible machining, shorter setup times and increased productivity possible. In other words, it makes it much easier to set up the machine.

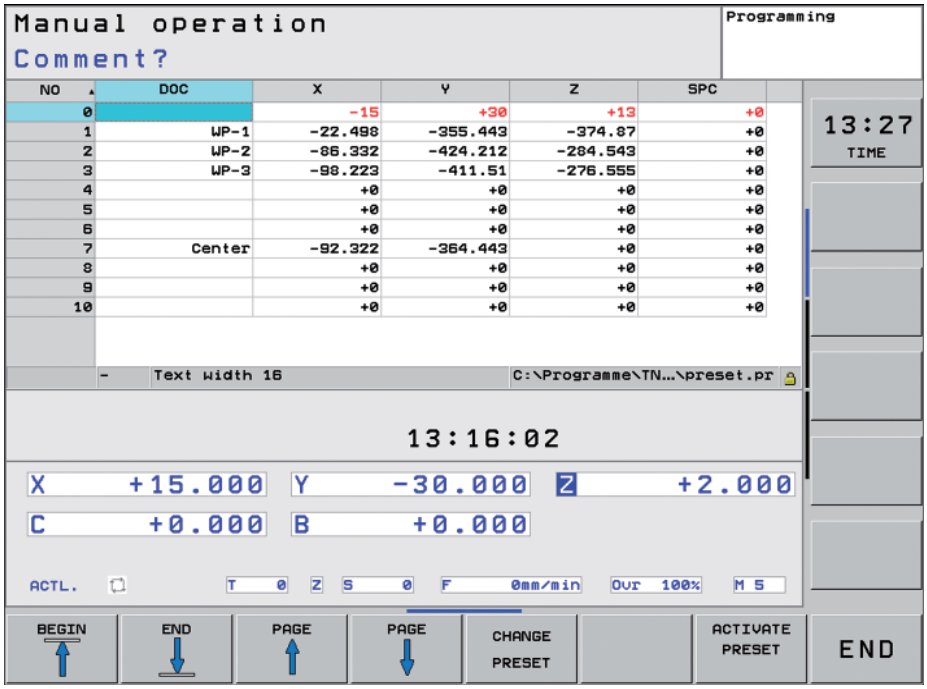
In the preset table you can save **any number of reference points** and assign an individual basic rotation to each one.

There are three ways to save reference points in the preset table:
- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Workpiece presetting
At a corner, for example, or in the center of a bolt hole circle





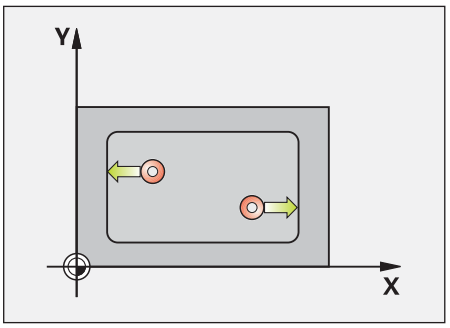


Minimize Setup and Nonmachining Time
– The TNC 320 Manages and Measures

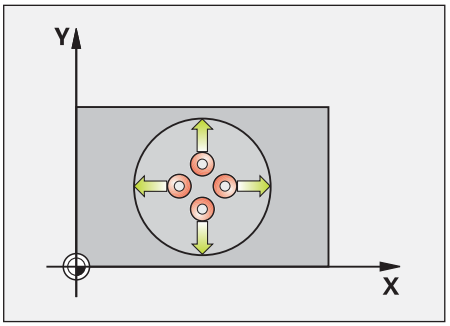
The difference in requirements placed on the classical machine for tool and mold-making and machining centers are becoming ever less distinct. Of course the TNC 320 is capable of controlling automated manufacturing processes. It has the functions required to manage and measure tools and inspect the machining process. It helps you reduce non-cutting time, increase productivity and improve production quality.

Inspecting workpieces for proper machining and dimensional accuracy
The TNC 320 features a number of measuring cycles for checking the geometry of the machined workpieces. To run the measuring cycles, you simply insert a touch probe from HEIDENHAIN (see page 24) into the spindle in place of a tool. This enables you to

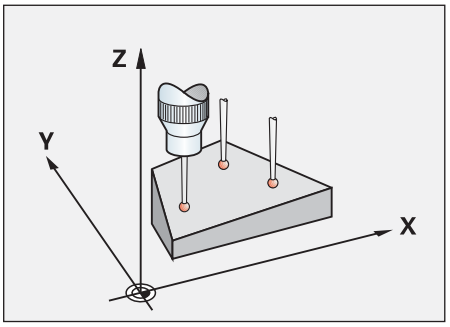
- recognize a workpiece and call the appropriate part program,
- check whether all machining operations were conducted correctly,
- determine infeeds for finishing,
- detect and compensate tool wear,
- and ascertain machining error trends.



Length measurement

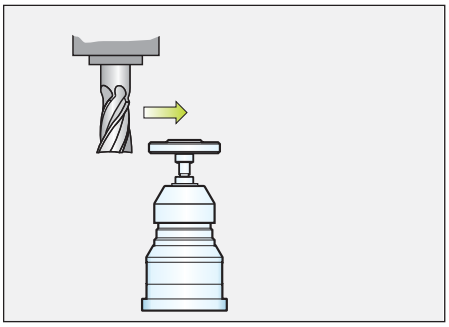


Circular pocket/hole measurement

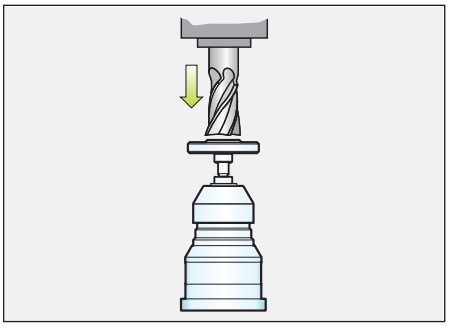


Measuring the angle of a plane

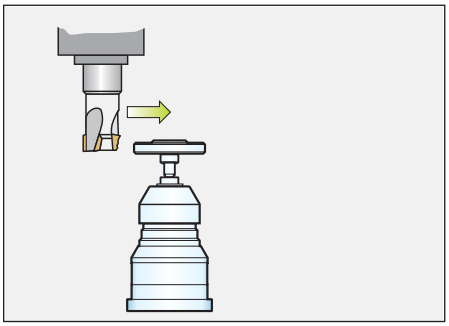
Workpiece measurement and automatic compensation of tool data
Together with the TT 140 touch probe for tool measurement (see page 25) the TNC 320 can automatically measure tools while they are in the machine. The TNC 320 saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool during machining you can quickly and directly measure wear or breakage to prevent scrap or rework. If the measured deviations lie outside the tolerances, or if the monitored life of the tool is exceeded, the TNC 320 locks the tool and automatically inserts a replacement tool.



Measuring the tool radius



Measuring the tool length



Measuring tool wear

Tool management
For machining centers with automatic tool changers, the TNC 320 offers a central tool memory for any number of tools. The tool memory is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 320 manage your tool names. The control prepares the next tool change while the current tool is still cutting. This significantly reduces the non-cutting time required for changing tools.

Machining with Four Axes
– Swivel Head and Rotary Table Controlled by the TNC (Option)

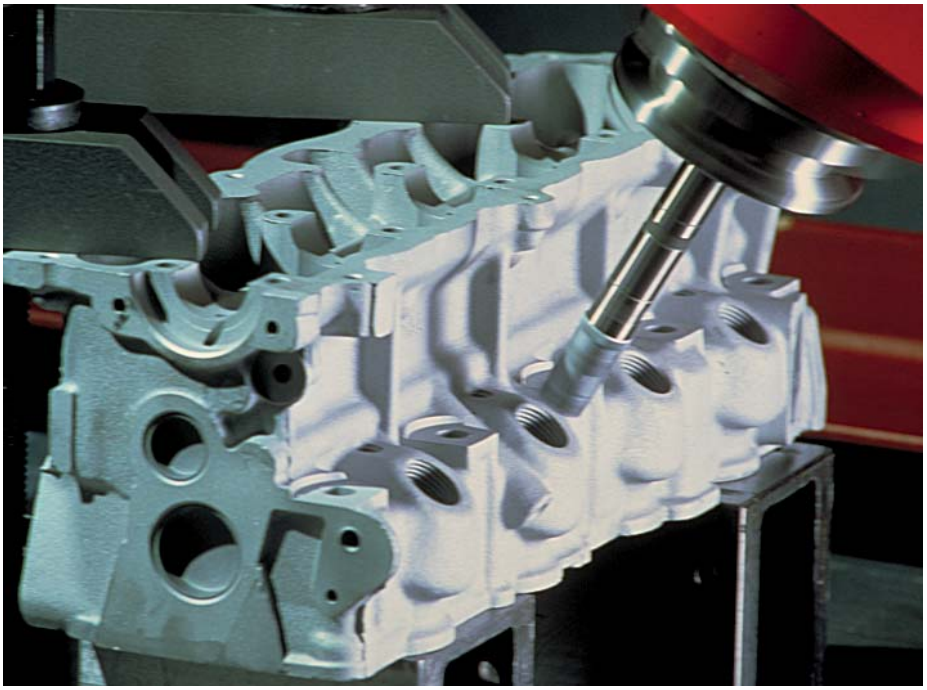
Many five-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply tilted about one or more rotary axes or wrapped onto a cylindrical surface. The TNC supports you with application-oriented functions to help you write and edit such programs quickly and simply without a CAD/CAM system.

Tilting the working plane*
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC 320 helps you to save a great deal of programming time.

You program the part as usual in the working plane (e.g. the X/Y plane), but it is machined in a plane that is rotated in one or more axes about the main plane.

The PLANE feature makes it easy to define a tilted working plane: You can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. Clearly arranged support graphics assist you during input.

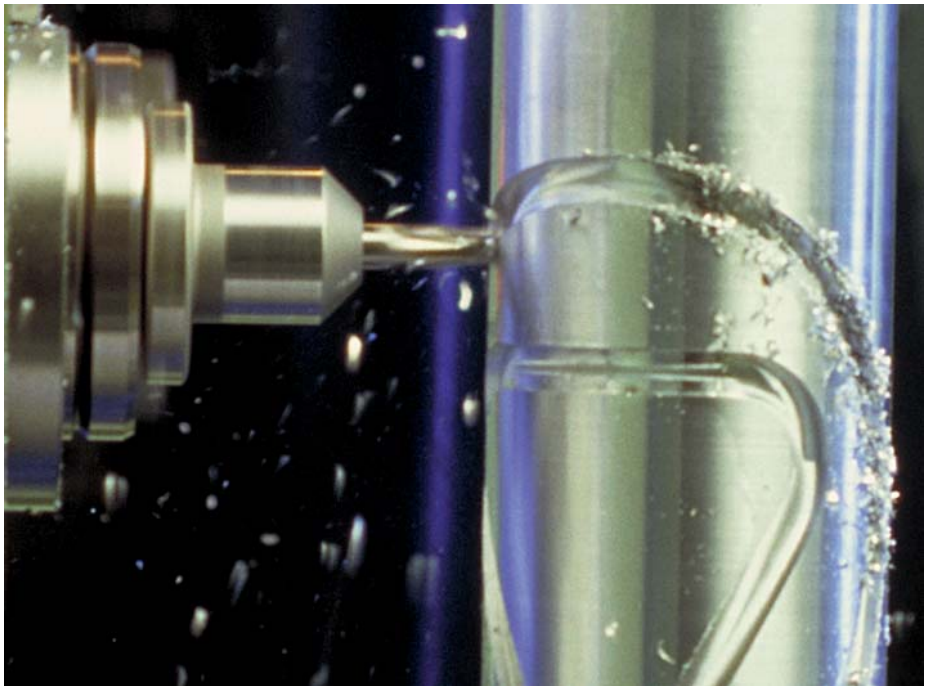
You can define the positioning behavior with the PLANE function so that there are no unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making everything that much easier.



Machining cylindrical surfaces*
With the TNC 320 it is quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane as if the cylinder surface were unrolled. You enter a contour in two dimensions—as if in a plane—and the TNC 320 then calculates and machines the corresponding cylindrical contour.

The TNC 320 features three cycles for cylindrical surface machining:
- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling

***** The machine must be prepared by the machine tool builder for this function.



Programming, Editing, Testing
– The TNC 320 Opens Endless Possibilities

The TNC 320 is just as universal in application as it is flexible in machining and programming.

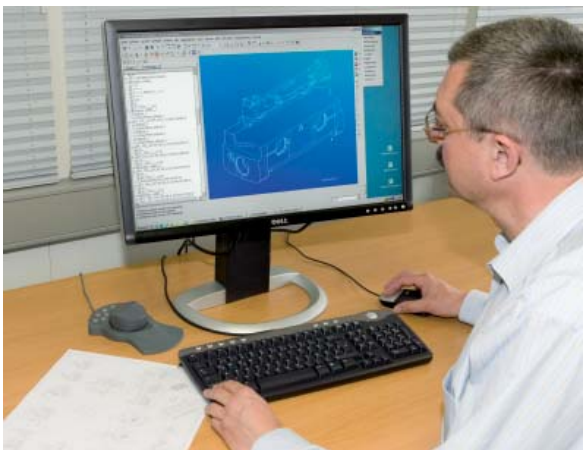
Positioning with Manual Data Input
You can start working with the TNC 320 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

Programming at the machine
HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. With **conversational programming** you can forget about memorizing G codes. Instead you use dedicated keys and soft keys to program line segments, circular arcs and cycles. With a keystroke, you initiate a HEIDENHAIN plain language dialog, and the TNC begins immediately to support you actively in your work. Unambiguous questions and prompts help you enter all the required information.

If you are used to **DIN/ISO programming,** however, the TNC 320 is still the right control for you. It displays soft-key rows dedicated to the most important DIN/ISO commands. Or you connect a USB keyboard and use it to write the program.

Whether plain-language prompts, dialog guidance, programming steps or soft keys, all texts are available in all numerous languages.

Creating programs offline
The TNC 320 is also well equipped for offline programming. It can be integrated through its interfaces into networks and connected with programming stations or other data storage devices. The TNC 320 can also run programs that were written in DIN/ISO format.



– Graphic Support in Any Situation

Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the TNC 320 draws every entered traverse command on the screen.

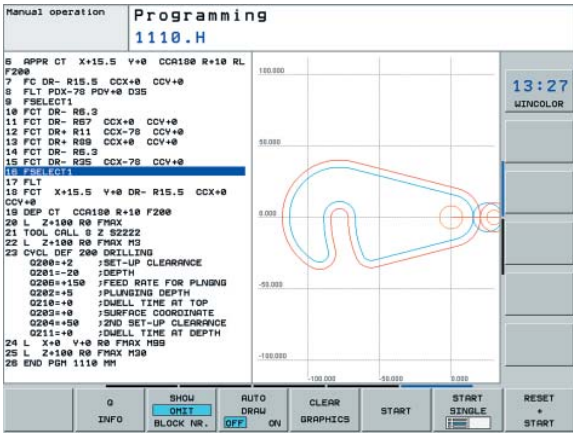
Program verification graphics
To play it safe before running a program, the TNC 320 can graphically simulate the machining of the workpiece. It can display the simulation in the following ways:

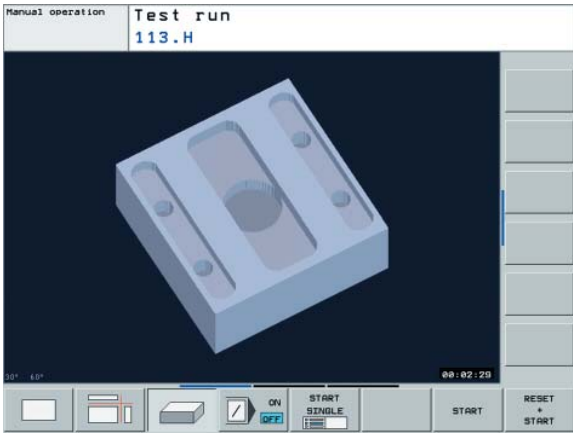
- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

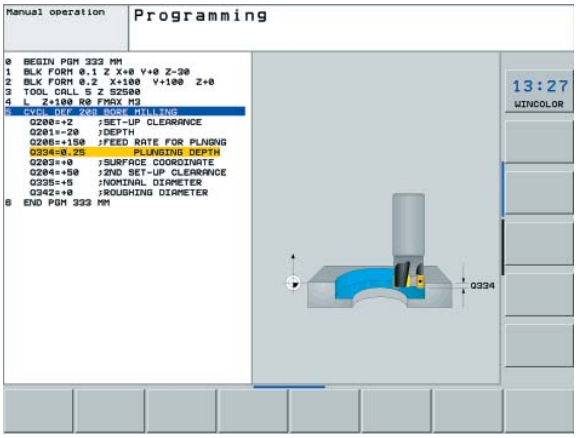
Details can be displayed in magnification. In addition, the TNC 320 indicates the calculated machining time in hours, minutes and seconds.

Program-run graphics
On the TNC 320, you can run the programming graphics or verification graphics even while the workpiece is being machined. Also, it shows a real-time graphic of the machining progress during program run. Coolant spray and protective enclosures usually obstruct any direct view of the actual workpiece. You can get around this with a simple keystroke to see the simulated progress of workpiece machining.

Help graphics
During cycle programming in the plain-language dialog, the TNC shows a separate illustration for each parameter. This makes it easier to understand the function and accelerates programming.







Programming in the Workshop
– Straightforward Function Keys for Complex Contours

Programming 2-D Contours

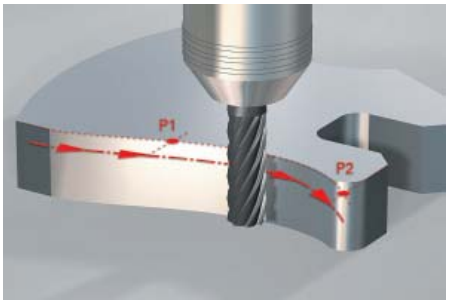
Two-dimensional contours are the daily bread of the modern machine shop. Here the TNC 320 offers a variety of possibilities.

Programming with path function keys
If contours are dimensioned for NC, which means that the end points are specified in Cartesian or polar coordinates, then you can program them directly with the path function keys.

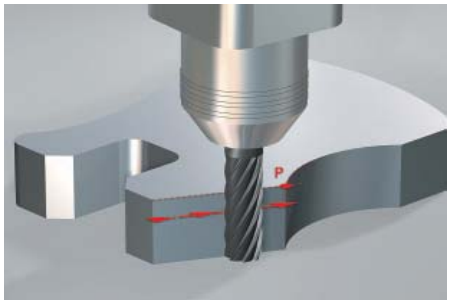
Straight and circular contour elements
To program a line segment, for example, simply press the key for linear traverse. The TNC 320 asks for all information required for a complete programming block, such as target coordinates, feed rate, cutter radius compensation and machine functions. Appropriate path function keys for circular movement, chamfers, and corner rounding simplify your programming. To avoid surface blemishes during approach or departure from the contour, it must be approached smoothly—that is, tangentially.

You simply specify the starting or end point of the contour and the approaching or departing radius of the cutter edge—the control does the rest for you.

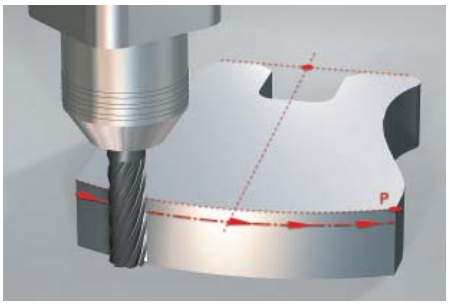
The TNC 320 can look ahead over a radius-compensated contour for up to 99 blocks to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.



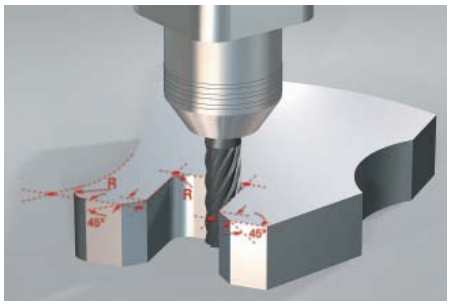
CT
Circular path defined by its end point, with a smooth (tangential) departure from the previous contour element



L
Straight line defined by its end point

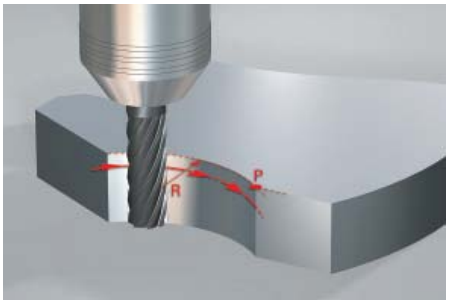


CC C
Circular path defined by its center point, end point, and rotational direction



RND Rounding: circular path defined by radius and corner point, with a smooth (tangential) transition to its adjoining contour elements

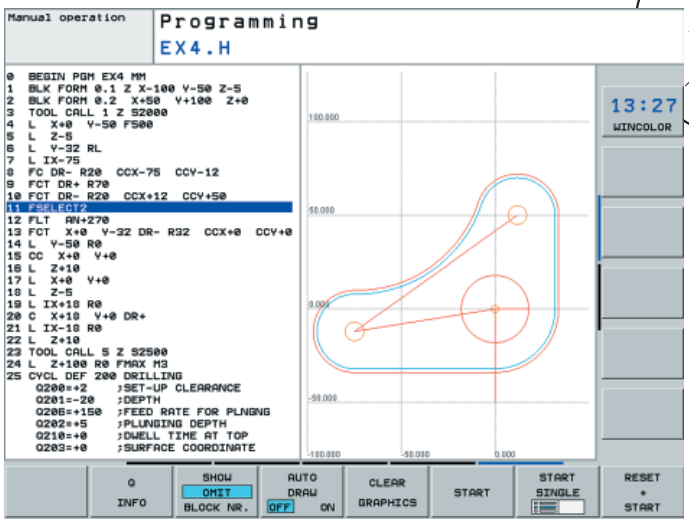
CHF Chamfer: defined by the corner point and the chamfer length

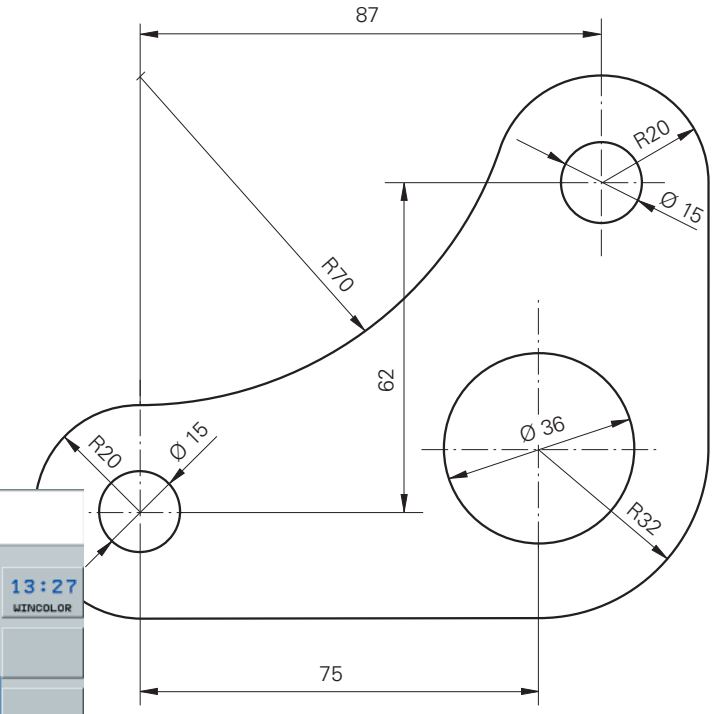


CR
Circular path defined by its radius, end point and rotational direction

– Programming Contours Unconventionally

FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control's free contour programming feature, in such cases you simply type in the known data—without first having to convert or calculate your data! It does not matter if individual contour elements are not completely defined as long as the complete contour has been. If the given data result in more than one mathematical solution, the helpful TNC 320 programming graphics present the possible variants for your selection.





Programming in the Workshop
– Field-Proven Cycles for Recurring Operations

Comprehensive Fixed Cycles for Milling, Drilling and Boring

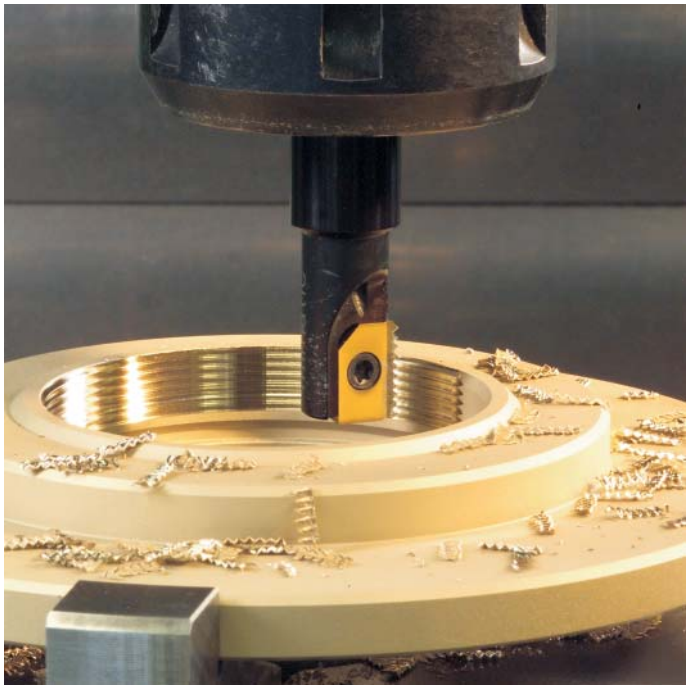
Frequently recurring operations that comprise several working steps are stored in the TNC 320 as cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

Standard cycles
Besides the fixed cycles for drilling, tapping (with or without floating tap holder), thread milling, reaming and boring, there are cycles for hole patterns and milling cycles for clearing plane surfaces, and for roughing and finishing pockets, slots and studs.

Cycles for complex contours
Subcontour List cycles (SL) are particularly helpful for clearing pockets with combined contours. This expression denotes fixed cycles for pilot drilling, roughing and finishing in which the contour or contour segments are defined in subprograms. In this way, one contour description can be used for more than one operation using different tools.

Up to twelve **subcontours** can be superimposed for machining. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Different components are combined to form a single pocket in which the tool avoids the islands.

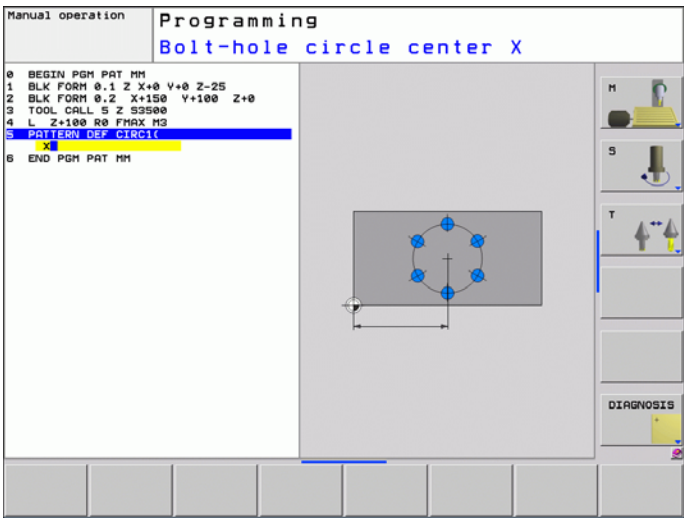
The TNC 320 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control identifies material remaining in inside corners so that it can be cleared later with smaller tools. A separate cycle is used for milling to the finished dimension.



OEM cycles
As original equipment manufacturers (OEMs), machine tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 320. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. CycleDesign enables you to organize the input parameters and soft-key structure of the TNC 320 to suit your own needs.

3-D machining with parametric programming
With parameter functions you can program simple 3-D geometric figures that can easily be described mathematically. Here you can use the basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions. Parametric programming also offers you a simple method of realizing 3-D operations for which there are no standard cycles. Of course, parametric programming is also suited for **2-D contours** that cannot be described with line segments or circular arcs, but rather through mathematical functions.

Stay simple and flexible when programming machining patterns
Machining positions are often arranged in patterns on the workpiece. With the TNC 320, you can program very diverse machining patterns simply and extremely flexibly—of course with graphic support. You can define as many point patterns as desired with various numbers of points.



Programming in the Workshop
– Reusing Programmed Contour Elements

Coordinate transformation
If you should need a contour that has already been programmed at another position or in a different size, the TNC 320 offers you a simple solution: coordinate transformation.

With coordinate transformation you can, for example, **rotate or mirror** the coordinate system, or **shift the datum**. With a **scaling factor** you can enlarge or reduce contours to respect shrinkage allowances or oversizes.

Program section repeats, subprograms, program calls
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal of programming time.

In **program section repetition,** you label a section of the program and during program run the TNC repeats the section successively as many times as required.

You can mark a program section as a **subprogram** and then call it at any point in the program and as often as you want.

With the **program call** function you can even use a completely separate program at any place in your current program. This gives you convenient access to pre-programmed, frequently needed working steps or contours.

Of course you can also combine these programming techniques.

– Fast Availability of All Information

Do you have questions on a programming step but your User's Manual is not at hand? No problem: The TNC 320 numerical control and TNC 320 programming station now feature TNCguide, a convenient help system that can show the user documentation in a separate window.

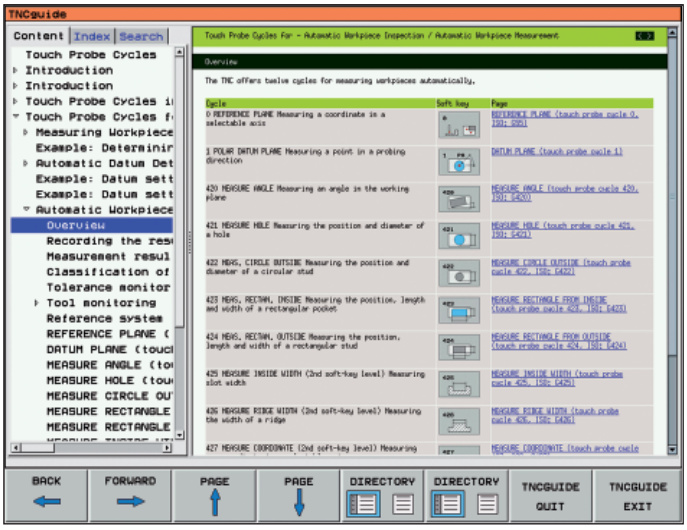
You can activate the TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key with a cursor in the shape of a question mark. You switch the cursor by simply clicking the help symbol ( ) that appears on all TNC screens.

TNCguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. This function is particularly helpful with the soft keys. The method and effect of operation is explained in detail.

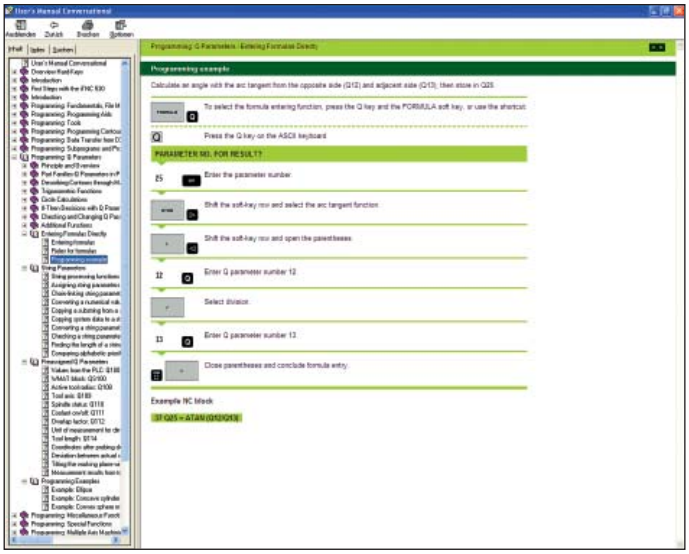
You can download the documentation in the desired language from the HEIDENHAIN homepage into the corresponding language directory on the TNC hard disk.

The following manuals are available in the help system:

- User's Manual for Conversational Programming
- User's Manual for Cycle Programming
- User's Manual for DIN/ISO Programming
- User's Manual for the TNC 320 Programming Station (only included in the programming station)



TNCguide integrated in the control, e.g. on the TNC 320 ...



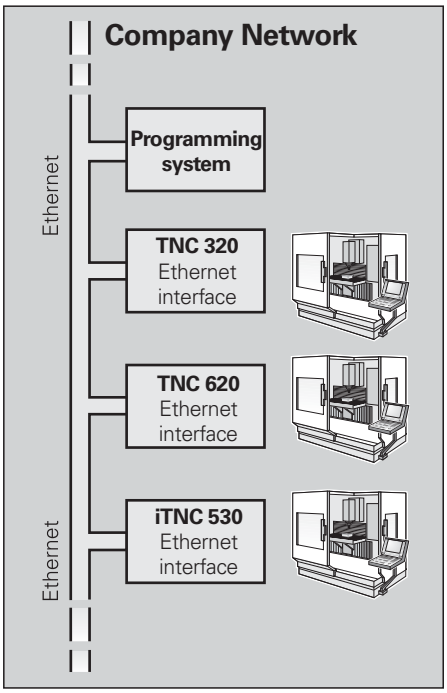
… or at the programming station

Open for Communication
– Fast Data Transfer with the TNC

The networked TNC 320
The TNC 320 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the TNC 320 features a latest-generation Fast Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 320 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 100 Mbps guarantees very short transfer times.

The transmitted programs are saved in the internal memory of the TNC 320 and are run from it at high speed.

For well-organized program management on your control, simply place the individual files in directories (folders). You can structure the respective directories through individual subdirectories.



Programs for data transfer
With the aid of the free PC program **TNCremoNT** from HEIDENHAIN and an Ethernet or other data interface you can
- transfer remotely stored part programs and tool or pallet tables bidirectionally
- make backups.

With the powerful **TNCremoPlus** software you can also transfer the screen contents of the control to your PC by means of the live-screen function.

– The TNC 320 Programming Station

Why a programming station?
It's well known that it is easy to create part programs on a TNC 320 at the machine, even while another part is being machined. Nevertheless, it can often happen that short reloading times and other machining tasks hinder any prolonged or concentrated programming work. With the TNC 320 programming station you have the capability to program just as you do at the machine, but away from the noise and distractions of the shop floor.

Creating programs
Programming, testing and optimizing HEIDENHAIN conversational or DIN/ISO programs for the TNC 320 with the programming station substantially reduces machine idle times. You need not adjust your way of thinking—every keystroke fits. On the programming station you program on the same keyboard as at the machine.

Testing programs created offline
Of course you can also test programs that were written on a CAD/CAM system. The various views of the program verification graphics help you to easily spot contour damage and hidden details.

Training with the TNC 320 programming station
Because the TNC 320 programming station is based on the same software as the TNC 320, it is ideally suited for apprentice and advanced training. The program is entered on the original keyboard unit. Even the test run functions exactly as it does on the machine. This gives the trainee the experience needed to enable him to safely operate the machine later.

Because it can be programmed with smarT.NC, in plain language, and in DIN/ISO, the TNC 320 programming station is very well suited in schools for TNC programming training.

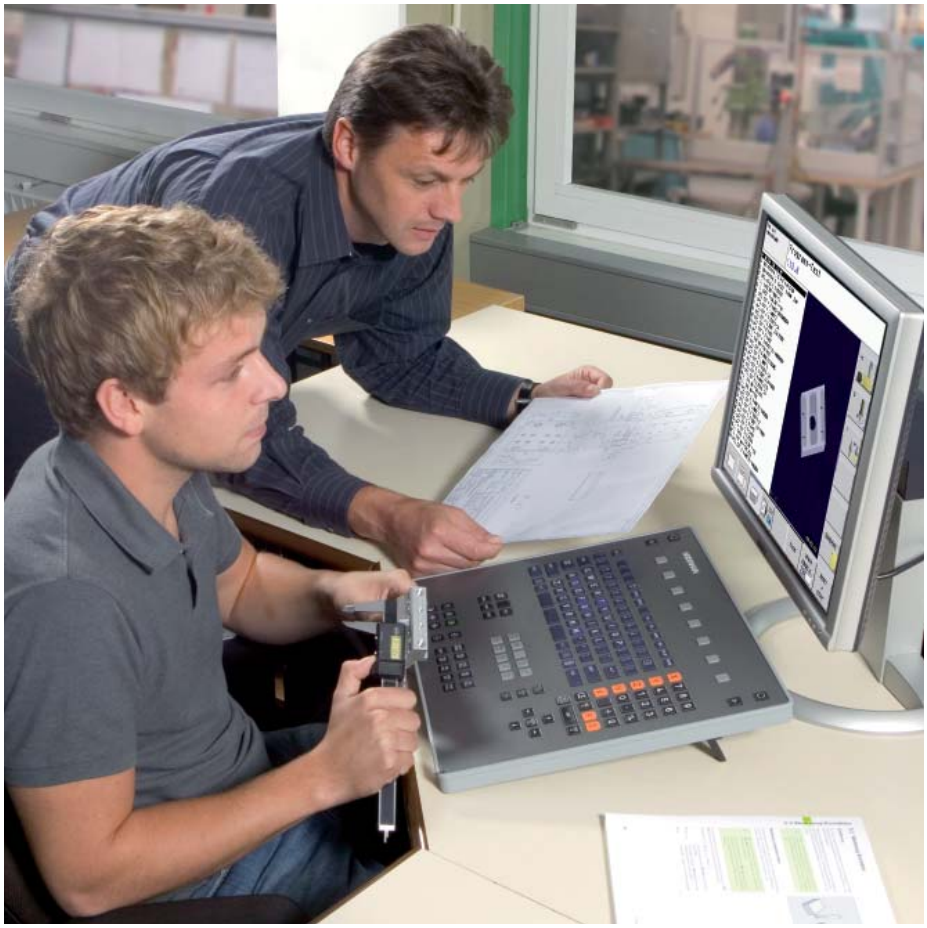
More information about the programming station and a free demo version is available on the Internet at *www.heidenhain.de.* Or simply ask for the *TNC 320 Programming Station* CD or brochure.

Your workstation
The programming station software runs on a PC. The PC screen shows you the TNC user interface as on the control, and offers the familiar graphic support. Depending on the version of the programming station, there are several types of possibilities for using it.

The free **demo version** contains all functions of the TNC 320, and permits short programs to be saved. It is programmed over the PC keyboard.

On the version with the TE 520 B **TNC operating panel** you create your programs as always, on a keyboard with the same function keys as the control on the machine. It also has a PC keyboard for G-code programming, file names and comments.

But you can also work without the TNC operating panel: a **virtual keyboard** simulating the TE appears on the PC screen with the TNC 320's most important dialog initiation keys.



Programming station with TNC operating panel

Workpiece Measurement
– Setup, Datum Setting and Measuring with Touch Trigger Probes

Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the TNC 320, touch probes can automatically perform setup, measuring and inspection functions.

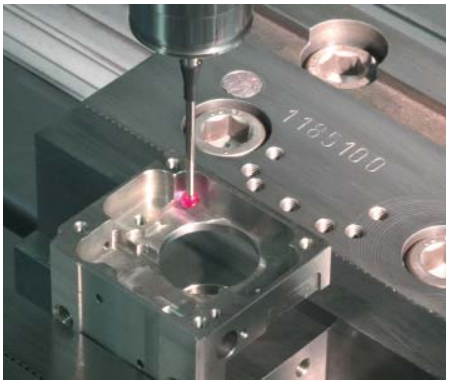
The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that, depending on the model, is transmitted either by cable or over an infrared beam to the control.

The touch probes* are inserted directly into the machine tool spindle. They can be equipped with various shanks depending on the machine. The ruby ball tips are available in several diameters, and the styli in different lengths.

* The touch probes must be interfaced to the TNC 320 by the machine tool builder.

Touch probe with **cable connection for signal transmission** for machines with manual tool change:
TS 220 – TTL version

Touch probes with **infrared signal transmission** for machines with automatic tool change:
TS 440 – Compact dimensions
TS 444 – Compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
TS 640 – Standard touch probe with wide-range infrared transmission
TS 740 – High probing accuracy and repeatability, low probing force



SE 640

TS 220

TS 640

TS 440

More information about workpiece touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure or CD.

Tool Measurement
– Measuring Length, Radius and Wear Directly in the Machine

The tool is of course a definitive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. A suitable touch trigger probe for tool measurement is the **TT 140*.** It is installed directly in the machine's workspace, where it permits tool measurement either before machining or during interruptions.

The TT 140 tool touch probe captures the tool length and radius. When probing the tool, either while rotating or at standstill (such as for measuring individual teeth), the contact plate is deflected and a trigger signal is transmitted to the TNC 320.

* The touch probe must be interfaced to the TNC 320 by the machine tool builder.

TT 140

More information about tool touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure or CD.

Positioning with the Electronic Handwheel
– Delicate Axis Traverse

To set up the workpiece you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more sensitive way, however, is to use the electronic handwheels from HEIDENHAIN.

You can move the axis slide through the feed motors in direct relation to the rotation of the handwheel. For delicate operations you can set the transmission ratio to certain preset distances per handwheel revolution.

HR 130 and HR 150 panel-mounted handwheels
The panel-mounted handwheels from HEIDENHAIN can be integrated in the machine operating panel or mounted at another location on the machine. An adapter permits connection of up to three HR 150 electronic handwheels.

HR 410 portable handwheel
If you need to get a closer look at the workpiece in the machine's working envelope, then ideally you should use the HR 410 portable handwheel. The axis keys and certain function keys are integrated in the housing, so you can switch axes and set up the machine at any time—regardless of where you happen to be standing.

The following functions are available:
- Traverse direction keys
- Three keys with preset feed rates for latched traverse
- Actual-position-capture key
- Three keys for machine functions to be defined by the machine tool builder
- Permissive buttons
- Emergency stop button

HR 410

Overview
– Specifications

Specifications	
Components	• Main computer with TNC keyboard and integrated 15.1-inch TFT color flat-panel display with soft keys
Operating system	• HEROS real-time operating system for machine control
Memory	• 300 MB (on compact flash memory card CFR)
Input resolution and display step	• Linear axes: up to 1 µm • Angular axes: to 0.001°
Input range	• Maximum: 99 999.999 mm or 99 999.999°
Interpolation	• Linear in 4 axes • Circular in 2 axes • Circular in 3 axes with tilted working plane • Helical: superimposition of circular and straight paths
Block processing time	• 6 ms (3-D straight line without radius compensation)
Axis feedback control	• Position loop resolution: Signal period of the position encoder/1024 • Cycle time of position controller: 3 ms
Range of traverse	• Maximum: 100 m
Spindle speed	• Maximum: 100 000 rpm (analog speed command signal)
Error compensation	• Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion • Stick-slip friction
Data interfaces	• RS-232-C/V.24 max. 115 kbps • Extended data interface with LSV2 protocol for remote operation of the TNC 320 over the data interface with the HEIDENHAIN software TNCremoNT or TNCremoPlus • 100BaseT Fast Ethernet interface • 3 x USB 1.1
Diagnostics	• Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	• Operation: +5 °C to +45 °C • Storage: –35 °C to +65 °C

Overview
– User Functions

User functions	Standard	Option	
Brief description	•		Basic version: 3 axes plus closed-loop spindle
		○	1st additional axis for 4 axes and open-loop or closed-loop spindle
		○	2nd additional axis for 5 axes and open-loop spindle
Program entry	•		In HEIDENHAIN conversational format and DIN/ISO over soft keys or USB keyboard
Position data coordinates	•		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	•		Incremental or absolute dimensions
	•		Display and input in mm or inches
Tool compensation	•		Tool radius in the working plane and tool length
	•		Radius-compensated contour look-ahead for up to 99 blocks (M120)
Tool tables	•		Multiple tool tables with any number of tools
Constant contouring speed	•		Relative to the path of the tool center
	•		Relative to the tool's cutting edge
Parallel operation	•		Creating a program with graphical support while another program is being run
Rotary table machining		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in mm/min
Contour elements	•		Straight line
	•		Chamfer
	•		Circular path
	•		Circle center point
	•		Circle radius
	•		Tangentially connecting circular arc
	•		Corner rounding
Approaching and departing the contour	•		Via straight line: tangential or perpendicular
	•		Via circular arc
FK free contour programming	•		FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	•		Subroutines
	•		Program section repeats
	•		Calling any program as subroutine
Fixed cycles	•		Cycles for drilling, pecking, reaming, boring, counterboring, conventional and rigid tapping
	•		Cycles for milling internal and external threads
	•		Rectangular and circular pockets
	•		Cycles for clearing level and inclined surfaces
	•		Multioperation machining of straight and circular slots
	•		Multioperation machining of rectangular and circular pockets
	•		Linear and circular point patterns
	•		Contour train, contour-parallel contour pocket
	•		OEM cycles (special cycles developed by the machine tool builder) can be integrated
Coordinate transformation	•		Datum shift, rotation, mirror image, scaling factor (axis-specific)
		8	Tilting the working plane, PLANE function

User functions	Standard	Option	
Q parameters Programming with variables	• • • • • •		Mathematical functions =, +, –, *, /, sin α, cos α, angle α of sin α and cos α, $\sqrt{a}$, $\sqrt{a^2 + b^2}$ Logical operations (=, = /, <, >) Calculating with parentheses tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point Functions for calculation of circles
Programming aids	• • • • • •		Calculator Complete list of all current error messages Context-sensitive help function for error messages TNCguide: The integrated help system. User information available directly on the iTNC 320 Graphical support for programming cycles Comment and structure blocks in the NC program
Actual position capture	•		Actual positions can be transferred directly into the NC program
Program verification graphics Display modes	• • •		Graphic simulation before a program run, even while another program is running Plan view / projection in 3 planes / 3-D view, also in tilted working plane Magnification of details
Programming graphics	•		In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while the blocks are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	• •		Graphic simulation during real-time machining Plan view / projection in 3 planes / 3-D view
Machining time	• •		Calculation of machining time in the Test Run operating mode Display of the current machining time in the Program Run operating mode
Returning to the contour	• •		Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining Program interruption, contour departure and return
Preset tables	•		One preset table for storing reference points
Datum tables	•		Several datum tables for storing workpiece-related datums
Touch probe cycles	• • • •		Touch probe calibration Compensating workpiece misalignment Datum setting, manual or automatic Automatic tool and workpiece measurement
Parallel secondary axes	• • •		Compensating movement in the secondary axis U, V, W through the principal axis X, Y, Z Including movements of parallel axes in the position display of the associated principal axis (sum display) Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
Conversational languages	•	 41	English, Chinese (traditional, simplified), Czech, Danish, Dutch, Finnish, French, German, Hungarian, Italian, Polish, Portuguese, Russian (Cyrillic), Spanish, Swedish For more conversational languages, see *Options*

– Options, Accessories

Option	Option number	As of NC software 340551-	ID	Comment
Additional axis	–	01	–	• 1st additional axis for 4 axes and open-loop or closed-loop spindle • 2nd additional axis for 5 axes and open-loop spindle
Software option 1	**8**	01	536164-01	**Rotary table machining** • Programming of cylindrical contours as if in two axes • Feed rate in mm/min
		03		**Interpolation** • Circular in 3 axes with tilted working plane **Coordinate transformation** • Tilting the working plane
		04		• PLANE function
Additional languages	**41**	04	530184-01 -02 -03 -04 -06 -07 -08 -09 -10	Additional conversational languages • Slovenian • Slovak • Latvian • Norwegian • Korean • Estonian • Turkish • Romanian • Lithuanian

Accessories	
Electronic handwheels	• One **HR 410** portable handwheel or • One **HR 130** panel-mounted handwheel or • Up to three **HR 150** panel-mounted handwheels via the HRA 110 handwheel adapter
Workpiece measurement	• **TS 220** touch trigger probe with cable connection or • **TS 440** touch trigger probe with infrared transmission or • **TS 444** touch trigger probe with infrared transmission or • **TS 640** touch trigger probe with infrared transmission or • **TS 740** touch trigger probe with infrared transmission
Tool measurement	• **TT 140** triggering touch probe
Software for PCs	• **TeleService:** Software for remote diagnosis, monitoring, and operation • **TNCdiag:** Software for fast and easy fault diagnosis • **CycleDesign:** Software for creating your own cycle structure • **TNCremoNT:** Software for data transfer—free of charge • **TNCremoPlus:** Software for data transfer with live-screen function
TNC 320 programming station	Control software for PCs for programming, archiving, and training • Full version with control keyboard • Full version with virtual keyboard • Demo version (operated via PC keyboard—free of charge)

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

www.heidenhain.de

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see www.heidenhain.de

DE **HEIDENHAIN Technisches Büro Nord**
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
08468 Heinsdorfergrund, Deutschland
☏ 03765 69544

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☏ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☏ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1345

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
E-mail: vicsales@fcrmotion.com

BA Bosnia and Herzegovina → **SL**

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **Belarus**
GERTNER Service GmbH
50354 Huerth, Germany
www.gertner.biz

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
02770 Espoo, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: service@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chennai – 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0073, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN LTD.**
Gasan-Dong, Seoul, Korea 153-782
www.heidenhain.co.kr

ME Montenegro → **SL**

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
E-mail: isoserve@po.jaring.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-489 Warszawa, Poland
www.apserwis.com.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500338, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
125315 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **Posredništvo HEIDENHAIN NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain-hubl.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
01133 Kiev, Ukraine
www.gertner.biz

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

VN **AMS Advanced Manufacturing Solutions Pte Ltd**
HCM City, Viêt Nam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za

Zum Abheften hier falzen! / *Fold here for filing!*

551 025-25 · 15 · 10/2009 · F&W · Printed in Germany